

**DEVELOPMENT OF A VORTICITY-VELOCITY NAVIER-STOKES
FORMULATION FOR THE STUDY OF COMPRESSIBILITY EFFECTS ON
DYNAMIC STALL**

1N-34
371994

K.N. GHIA*

Department of Aerospace Engineering and Engineering Mechanics

U. GHIA*

Department of Mechanical, Industrial and Nuclear Engineering

University of Cincinnati
Cincinnati, Ohio 45221

This research was supported by NASA Ames Research Center, Moffett Field, California, under Cooperative Agreement No. NCC2-5096, with Drs. Dochan Kwak and Stuart E. Rogers as Technical Monitors. Grant period September 1, 1994 - February 29, 1996

Distribution of this report is unlimited.

* Professor.

July 1996

ABSTRACT

The research performed under NASA Ames Cooperative Agreement NCC2-5096 between September 1, 1994 and February 29, 1996 is reviewed. Two areas of research were pursued.

The first major area of this study was to develop a vorticity-velocity formulation and numerical solution algorithms suitable for the analyses of incompressible as well as low-to-moderate-speed compressible flows. Research performed towards contributing to the determination of the appropriate vorticity and dilation creation boundary conditions suggested to temporarily set aside this approach and use a primitive-variable approach other than the pseudo-compressibility approach used by Drs. Kwak and Rogers. The second major area of study was initiated to comprehensively examine the INS-2D and INS-3D programs from the point of view of boundary conditions. The research carried out was documented in the form of two technical papers which are included in Appendices A and B; the boundary-condition related issues for INS-3D are briefly mentioned.

ENHANCEMENTS ACHIEVED FOR INS-3D

A critical assessment of the suitability of the public domain unsteady Navier-Stokes code INS-3D (incompressible Navier-Stokes 3D) for dynamic motion CFD was undertaken. Dynamic motion CFD focuses upon simulating the strongly nonlinear unsteady effects that are associated with bodies (typically aircraft) executing rapid maneuvers. Theoretically, such problems are boundary driven, and the boundary condition analyses and their implementation in code plays a dominant role in the accuracy and validity of the results generated. The examination of results generated using the boundary conditions supplied with INS-3D for the problem of the formation of a dynamically induced stall vortex by the rapid pitch-up of a NACA 0015 wing has revealed several areas needing improvements in the existing analysis. Typically, the strength of the resulting dynamic stall vortex is over-predicted, and the accuracy of the prediction of its location and strength with time exceeds uncertainty estimates for both the computed and experimental values. This has led to the critical assessment of some of the boundary conditions provided in INS-3D and the modification of some of these conditions.

As supplied, the inflow-outflow boundary condition analysis for the far field flow employs a method-of-characteristics (inviscid) analysis to determine the far-field pressure distribution. It has been demonstrated that this inviscid analysis is somewhat inconsistent with a viscous flow code and, in the present case, leads to the existence of an unphysical pressure discontinuity along the far-field boundary at the locations where the flow switches from being inflow to being outflow. This inflow-outflow boundary condition has been replaced with a Navier-Stokes (viscous flow) consistent boundary condition analysis for the far-field flow. Smooth velocity and pressure variations have been demonstrated along the far-field boundary using the new analysis.

As supplied, INS-3D employs an averaging technique to implement all internal flow-through grid-induced boundaries. Such boundaries are either the boundaries of multiple grid blocks, or Chimera grid boundaries, or C-grid boundaries. As a result, the governing Navier-Stokes equations are not solved along such locations, rather nearby values are

averaged to inject a solution upon these points. This is inconsistent and results in disturbances being unphysically generated at such locations. A modification of the existing interpolation databases has been accomplished, which allows the solution of the governing Navier-Stokes equations along C-grid and spatially periodic boundaries. The ability to handle spatial periodicity is a new capability that has been added to INS-3D by this corrective mechanism. Consistent treatment of multiple grid block interfaces and Chimera grid boundaries still remains to be done.

As supplied, INS-3D employs pressure extrapolation to the body surface as its wall boundary condition. This is equivalent to the boundary-layer approximation of zero normal pressure gradient near body surfaces. Unfortunately, separated flow destroys the boundary layer, making zero normal pressure gradient an inaccurate approximation for the present class of strongly separated flows. Others have linked the normal pressure gradient to body acceleration for maneuvering body simulations. However, the present analysis indicates that both of these approaches are inadequate to accurately predict the strength and location of the dynamic-stall vortex during rapid maneuvers. Its location and strength directly impact the force and moment distributions acting on the vehicle and, in the present circumstances, these forces and moments are very strong and directly impact the pilot's controllability of the maneuver. Consequently, a theoretical investigation must be undertaken, focusing on the proper formulation of the wall boundary constraint condition. It should be noted that, for incompressible codes, the issue is the prediction of the surface pressure distribution, while for compressible codes, the analogous issue is the prediction of the surface density distribution.

Proceedings of the
First AFOSR Conference on
Dynamic Motion CFD

Edited by

L. SAKELL

*Air Force Office of Scientific Research (NA)
110 Duncan Avenue, Suite B-115
Bolling AFB, DC 20332-0001*

D. D. KNIGHT

*Department of Mechanical and Aerospace Engineering
Rutgers University - The State University of New Jersey
PO Box 909 · Piscataway, New Jersey 08855-0909*

Organized by and Hosted at

Rutgers University - The State University of New Jersey
New Brunswick, New Jersey

June 3-5, 1996

Appendix A

**Unsteady Three-Dimensional Navier-Stokes
Simulation of Dynamic Stall With Emphasis on
Boundary Conditions**

G. Osswald, K. Ghia, U. Ghia

University of Cincinnati

Unsteady Three-Dimensional Navier-Stokes Simulation of Dynamic Stall with Emphasis on Boundary Conditions[†]

G. A. Osswald*, K. N. Ghia*, and U. Ghia**

Computational Fluid Dynamics Research Laboratory

*Department of Aerospace Engineering and Engineering Mechanics

**Department of Mechanical, Industrial, and Nuclear Engineering
University of Cincinnati, Cincinnati, OH 45221

Abstract

Issues concerning the implementation of body surface, sub-domain interface, and far-field boundary conditions for the full unsteady viscous compressible or incompressible Navier-Stokes equations are addressed in the context of the dynamic-stall application. Indications are presented that the standard boundary condition analyses employed with many public-domain off-the-shelf Navier-Stokes results are both overdetermined and inconsistent with the unsteady viscous flow equations.

Introduction

Why the emphasis on boundary conditions? Because the only difference between an aircraft in flight and the flow within an air-conditioning duct is the boundary conditions. Both flows exactly satisfy the Navier-Stokes equations everywhere else. Stated alternatively, any systematic error in a boundary condition analysis is transmitted directly to the solution and is completely transparent to a truncation-error analysis. In the end, one will have obtained a good mathematical solution, but to an entirely different problem.

So how does a systematic error enter a boundary condition analysis? The answer is very subtle, not easily recognized, but steps to the very heart of theoretical fluid dynamics.

The remainder of the paper uses the dynamic stall problem of a rectangular wing with a NACA0015 cross-section (hereafter referred to as NACA0015 airfoil with winglets) maneuvering in pitch to high angles of attack, with its ensuing unsteady separation, to illustrate the point. Discussion will progress from the body surface boundary conditions to the far field inflow/outflow boundary typical of external flows.

Model Geometry

Figure 1 illustrates a three-dimensional instantaneous pressure solution for a maneuvering NACA0015 airfoil with winglets. The four-to-one aspect ratio matches the wind-tunnel artifact which used splitter plates (the winglets) to isolate the span from wind-tunnel wall effects. The simulation employs spatial periodicity in the spanwise direction, implemented by solving full Navier-Stokes equations within the spanwise periodic planes. Further, a replicated O-grid topology along the spanwise direction is used with full Navier-Stokes equations solved along the O-grid periodic boundary. The inflow/outflow boundary is located some 25 chords away from the body. Figure 2 illustrates spanwise-vorticity contours for the same instant, some two characteristic times after start of the motion. The Reynolds number, Re , is 45,000 based on free stream velocity and chord. The simulation is a "coarse-grid" or "engineering" direct numerical simulation with no turbulence

[†] This research is supported, in part, by AFOSR Grant No. F49620-92-J-0292, NASA Grant No. NCC2-5096, and Ohio Supercomputer Grant No. PES070-5.

modeling. The solution of the unsteady viscous Navier-Stokes equations is subject solely to the boundary conditions.

Body Surface Boundary Conditions

No-slip applies at the body surface. However, in addition to wall temperature, a compressible solution requires surface density, while an incompressible solution requires surface pressure. Herein lies the subtlety. Anything short of full unsteady conservation of mass, linear and angular momentum, and energy, with all terms active in the boundary analysis for surface density or pressure, will be inconsistent with the interior full unsteady viscous Navier-Stokes solution. Furthermore, such an inconsistency in the boundary analysis cannot be removed by grid refinement and cannot be seen by a truncation error analysis of the interior solution. To illustrate, it is customary in several public-domain results to invoke the flat plate boundary-layer approximation of either zero normal pressure gradient or, for the dynamic stall problem, a normal pressure gradient related solely to the normal acceleration of the body surface. Such an analysis drops the normal viscous contribution v_{yy}/Re entirely, and no amount of grid refinement can reintroduce the neglected term. Unfortunately, at a separation point, the only active term is v_{yy}/Re and the full unsteady viscous Navier-Stokes solution from the interior understands that this should have been the case. The reduced boundary analysis is inconsistent with the governing equation suite, and breaks the physics of the evolving flow. Comparing Figs. 3 and 4 illustrates the possible magnitude of this subtlety. Figures 3 and 4 represent midspan pressure contours for the NACA0015 airfoil-winglet combination two characteristic times after an impulsive start from rest at the fixed angle of attack of ten degrees. Figure 3 employs the zero normal pressure gradient boundary condition (appropriate for the fixed angle of attack calculation presented here) while Fig. 4 employs the full viscous equation suite to evolve surface pressure. Note the 400% difference in minimum surface pressure at the

trailing edge within the viscous separation zone and the non-physical, nearly-right-angle corners in the pressure contours near the body surface within Fig. 3. As a consequence, the formation, strength and instantaneous locations of the unsteady separation, and subsequently, the corresponding vortices, which dominate the dynamic stall problem, can be considerably altered by the reduced boundary analysis.

The Issue of Artificial Dissipation

The unsteady viscous Navier-Stokes equations understand viscosity quite well. They correctly admit temporal instability mechanisms as a solution. Under the right flow conditions, disturbances grow but not without bound. This is the critical mechanism of transition and the turbulent response of fluid systems. The unsteady viscous Navier-Stokes equations are, and can correctly be, locally unstable. Any attempt to falsely "stabilize" their response either by adding explicit artificial viscosity directly, through flux-vector splitting schemes of the convective terms, and/or turbulence modeling, suppresses the exact physical mechanisms that are responsible for the unsteady effects.

The smearing of the pressure contours at the trailing edge of the winglet can be clearly seen in Fig. 1. This is a direct consequence of the excessive dissipation injected by flux-vector splitting upon a viscous flow solution and represents unphysical behavior at such a trailing edge location. In principle, the viscous Navier-Stokes equations should not require the extra help of an inviscid artifact designed to rapidly damp oscillations near high-gradient regions. This high-gradient region should generate flow instability mechanism critical to the physical solution. Such subtle artificial suppression of an otherwise viable flow instability mechanism can completely alter the physics of the evolving unsteady flow.

Far-Field Boundary Conditions

Many public-domain compressible and incompressible Navier-Stokes simulations employ a method-of-characteristics analysis at an inflow or an outflow boundary. This represents a reduced boundary condition analysis, very similar to the boundary-layer analysis discussed above, except that now all of the viscous terms are artificially removed a priori. Unfortunately, the unsteady viscous Navier-Stokes equations are a parabolic system of equations owing to diffusion and always possess one imaginary characteristic, while the unsteady Euler equations are hyperbolic and possess all real characteristics. The characteristics of the unsteady Euler equations are not the characteristics of the unsteady viscous Navier-Stokes equations and any use of a method-of-characteristics analysis to set the boundary conditions for the unsteady viscous equations inconsistently overdetermines the boundary. The consequence for the present model geometry of a method-of-characteristic based inflow/outflow boundary analysis is demonstrated by Fig. 5 which plots the pressure distribution along the inflow/outflow boundary near the point on the O-grid topology where the flow switches from inflow to outflow. This is from the midspan plane within the solution shown in Fig. 1 and 2. The single grid-point discontinuity in pressure which occurs at the point where the flow switches from inflow to outflow is evident very clearly. Such a result is a physical impossibility for an unsteady viscous solution, but is a direct artifact of the method-of-characteristics. The interior Navier-Stokes operator smears this boundary-imposed discontinuity so that, near the body, its presence is difficult to see. However, pressure does have global impact. In the numerics = physics terminology, the physics side of the equation has just been broken.

Symptoms of Inconsistent Boundary Conditions

In general, the inability to reduce the residuals of all governing equations to machine zero everywhere within the flow domain is a clear indicator of an inconsistency somewhere within the analysis, and very often this inconsistency resides within the boundary condition analysis. It is impossible to simultaneously satisfy the interior difference equations and inconsistently overspecified boundary conditions. The numerics respond by saturating the residuals after dropping only a few orders of magnitude, thus violating the Navier-Stokes equations everywhere by redistributing the inconsistency amongst all interior equations. If one is lucky, the nonphysical behavior still manifests itself near the boundary. However, the tendency is to treat the symptoms rather than the cause. Thus, artificial dissipation is added near boundaries with the express intent of suppressing near boundary oscillations. High-order difference schemes are "required" to reduce their order of accuracy near boundaries, again with the express purpose of damping near-boundary oscillations. Flux-split differences appear within viscous algorithms. Excessive damping hides the behavior but at the expense of completely altering the physics of unsteady viscous flows. The physics of unsteady viscous flows is real temporal instability which amplifies disturbances, leading to persistent near-wall and not so near-wall unsteadiness, through the growth of coherent organized rotational motion in the guise of viscous eddies and unsteady separation. In summary, it is, absolutely essential that all elements of the boundary-condition analysis be consistent with the flow physics.

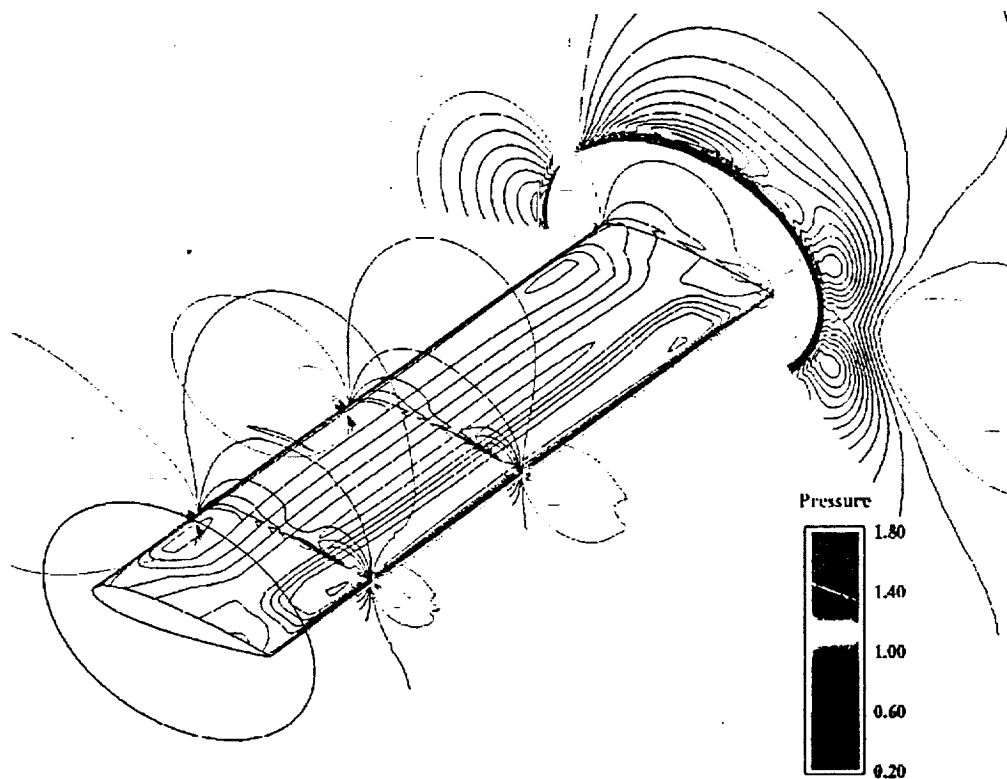


Figure 1) Pressure Contours for NACA0015 Airfoil with Winglets. $Re = 45,000$. $t = 2.0$

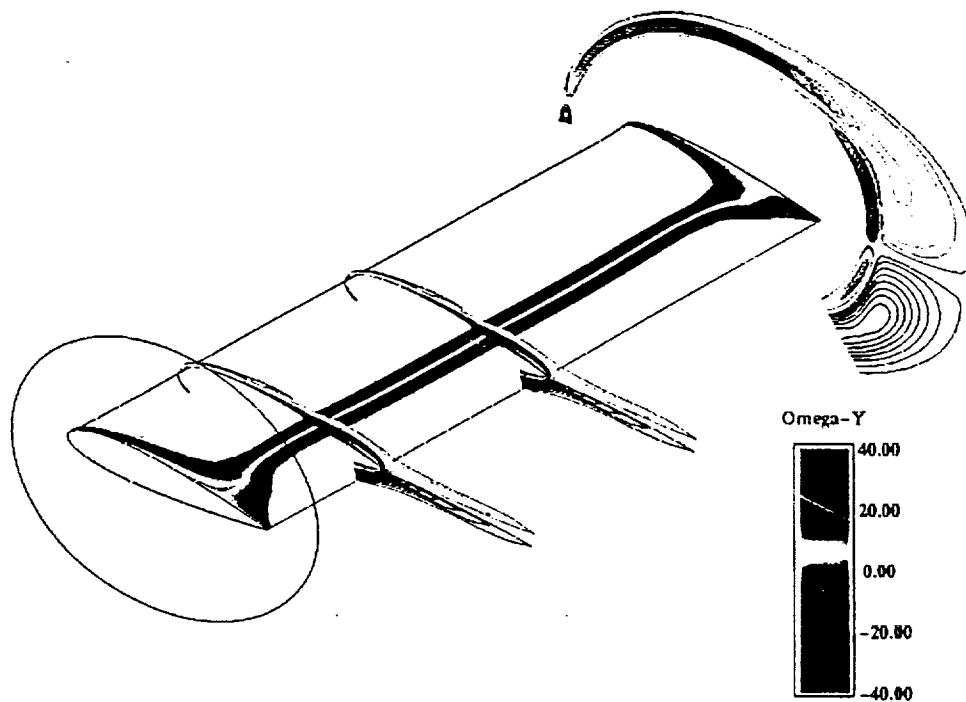


Figure 2) Spanwise Vorticity Contours for NACA0015 Airfoil with Winglets. $Re = 45,000$. $t = 2.0$

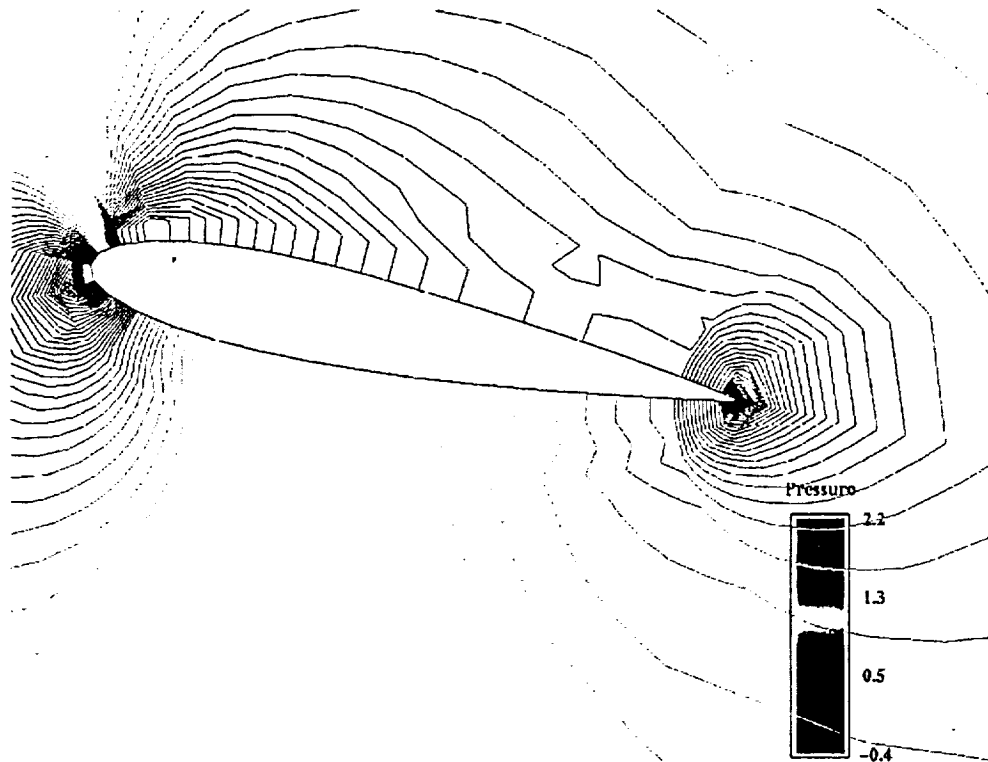


Figure 3) Midspan Pressure Contours. $\alpha = 10$ deg. (fixed)
Zero Normal Pressure Gradient Surface Boundary
Condition. $t = 2.0$

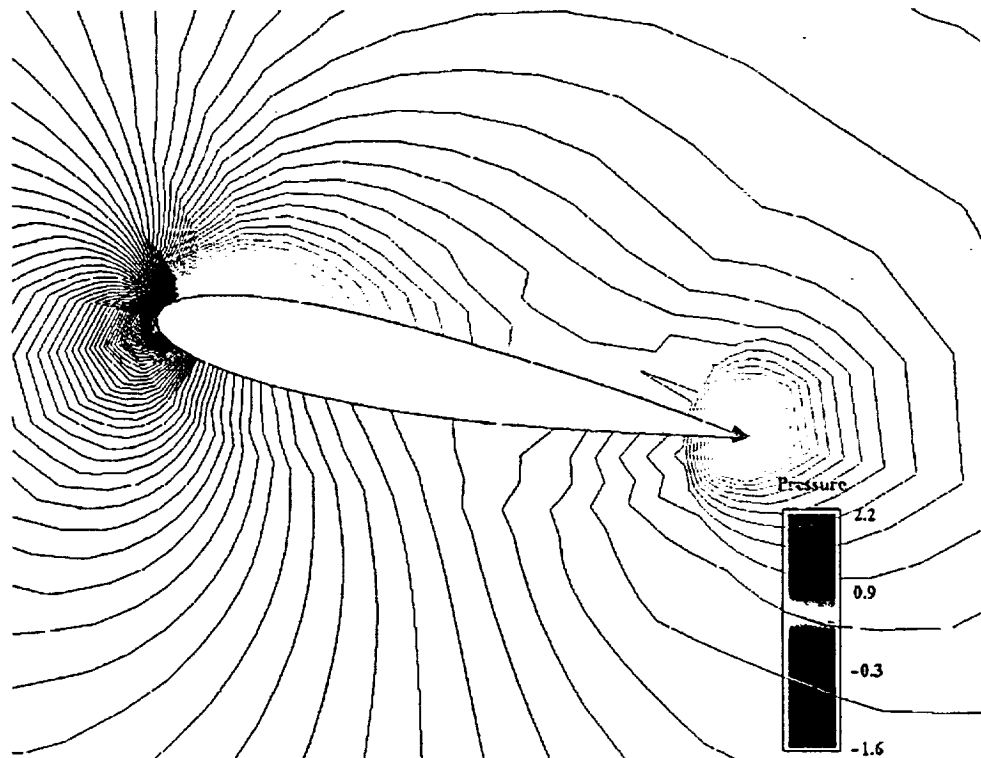


Figure 4) Midspan Pressure Contours. $\alpha = 10$ deg. (fixed)
Full Conservation Surface Boundary Condition.
 $t = 2.0$

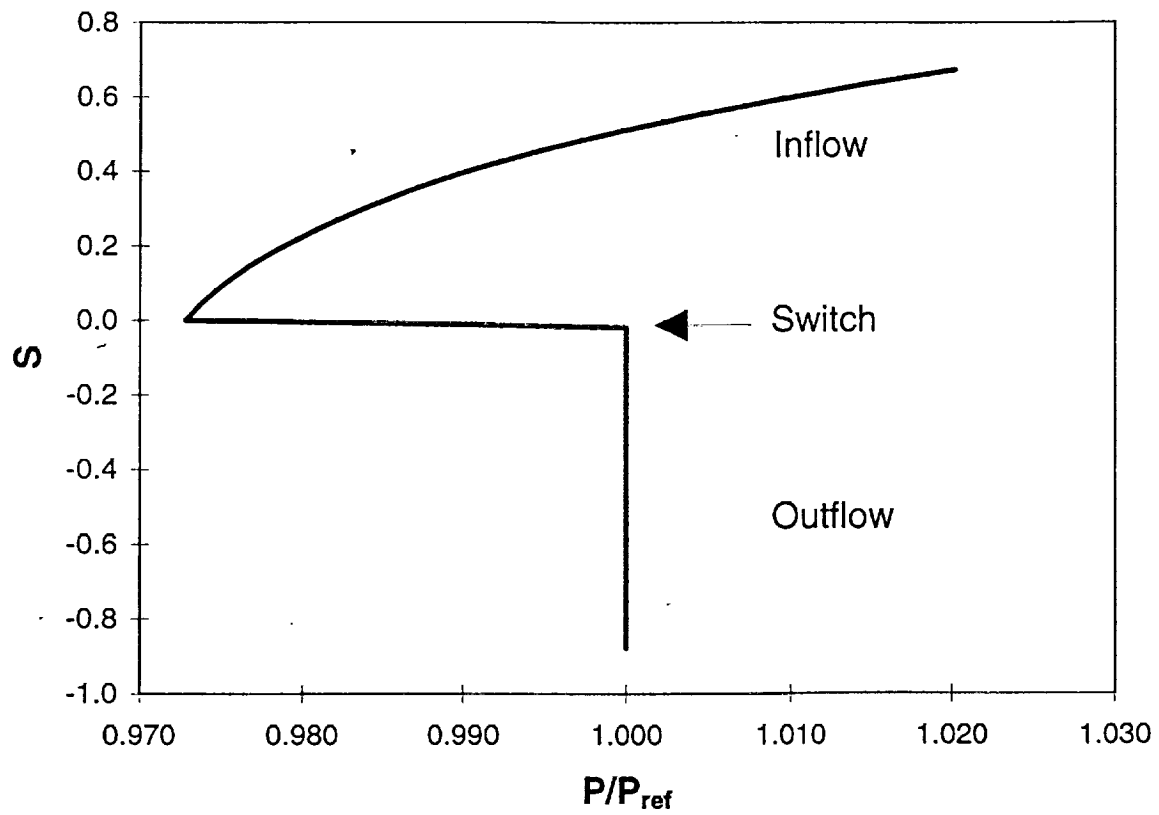


Figure 5) Pressure Distribution along O-Grid Inflow/Outflow Boundary at Midspan Near Section where Flow Switches from being Inflow to being Outflow.
 $Re=45,000$ $t=2.0$

Appendix B

**A Generalized Time-Dependent Analysis
Implemented on a Parallel Computer
for Studying Compressibility Effects on Dynamic Stall**

C. Noll, K. Ghia

University of Cincinnati

A Generalized Time-Dependent Analysis Implemented on a Parallel Computer for Studying Compressibility Effects on Dynamic Stall[†]

C. Noll and K. N. Ghia

Computational Fluid Dynamics Research Laboratory
Department of Aerospace Engineering and Engineering Mechanics
University of Cincinnati, Cincinnati, OH 45221

Abstract

This study has been undertaken to analyze the effect of compressibility on the dynamic-stall phenomenon by accurately simulating the prevailing mechanisms of the formation of the stall vortex. Towards this, a generalized analysis is developed in a time-dependent curvilinear coordinate system. It is implemented with flow-adaptive gridding and arbitrarily maneuvering and deforming bodies. One of the major contributions of this analysis is the elimination of the boundary condition of zero normal pressure gradient at solid surfaces, which breaks down in regions where separation occurs and in regions of high surface curvature. The branch cuts are treated properly by solving the complete Navier-Stokes equations at these interior points. The preliminary verification study led to the conclusion that computer resources on the CRAY Y-MP 864 are not adequate for the analysis being undertaken. This resulted in developing an object-oriented linear algebra class library for high-performance computation on parallel machines. At the time of the conference, the verification analysis was not completed, and as such, it was decided to provide some information towards the approach used in this work.

Introduction

Dynamic stall occurs when an airfoil is pitched rapidly past its static stall angle. The lift as well as the drag continues to increase, to a

dramatic extent, past the maximum possible static value. As seen in the experimental results of Jumper et al. (1988) in Fig. 1, as the angle of attack approaches 30° during the pitch-up cycle, there is a dramatic loss in lift, and a corresponding and sudden 'nose-down' moment. This event, termed dynamic stall, is a barrier technology, limiting the operational envelope of supermaneuverable aircraft and helicopters, and the performance of wind turbines and turbomachinery.

As is understood presently, the event of dynamic stall for incompressible flow is dominated by vortex interactions and instabilities. The instability mechanisms observed in two-dimensional flows will be described below. As the airfoil is accelerated from rest in a quiescent flow, the initial instability of this flow occurs when the Reynolds number (Re) exceeds a certain relatively small value and flow separates from the airfoil near the trailing edge. As the Reynolds number is increased further, this initial instability can amplify small asymmetric disturbances in the flow and an asymmetric instability occurs near the trailing edge, eventually developing into a Karman vortex street. This is a stable periodic flow for low enough Re . Due to the periodic nature of this flow, the specific point at which the airfoil is given an initial angular acceleration to begin the pitch-up is extremely important. The development of the resulting flow and the corresponding lift, drag and moment curves are observed to be very dependent on when the pitch-up is begun.

[†] This research is supported, in part, by AFOSR Grant No. F49620-93-1-0393, NASA Grant No. NC22-5096 and by Ohio Supercomputer Center Grant No. PES070-5.

For flows with moderate Re values, an instability occurs during the pitch-up maneuver near the leading edge where the boundary-layer flow separates, eventually intensifying into an attached counter-rotating upper-surface vortex. This vortex is the cause of the increased performance of the airfoil, in terms of its lift. Momentarily, it remains attached over the center of lift, resulting in very little change in moment applied to the airfoil. An underlying paired vortex forms on the airfoil surface. As the maneuver continues, an additional pair of vortices forms below this one. When the second vortex of this new pair is formed, the initial secondary vortex (paired with the dynamic-stall vortex) erupts out into the nearly-inviscid vortex flow, cutting off the feeding shear layer emanating from the leading edge, and the dynamic stall vortex evolves as shown in Fig. 2. The shear layer undergoes a Kelvin-Helmholtz instability, rolling up into the detached vortex which convects downstream over the airfoil, resulting in the large 'nose-down' moment on the airfoil. The dynamic-stall phenomenon in incompressible flow has been simulated very satisfactorily by K. Ghia, Yang, Osswald and U. Ghia (1992) and a suitable modulated suction/injection control law was devised by Yang, K. Ghia, U. Ghia and Osswald (1993).

The presence of compressibility, three-dimensionality, transition and turbulence, as well as a number of other factors, can significantly affect the resulting flow evolution. Compressibility can have significant effects on the sequence of instabilities described above. For flow with a relatively low free-stream Mach number, interaction with a rapidly maneuvering airfoil locally causes the flow to accelerate to a Mach number of two and higher, resulting in significant compressibility effects, specifically the production of additional vorticity. Maneuvers performed in supersonic free-stream flows have even more complex flow physics. In either case, as shown in Fig. 3, shocklets formed from local accelerations of the flow around vortices and wavy unstable shear layers can significantly affect the resulting flow structure according to the results of Chandreshekar et al. (1995). These shocklets exist over only a small range of angles of attack during

the maneuver. Small, temporally evolving shocks forming in the presence of vortex interactions and instabilities present challenging flow physics to be simulated by the computational fluid dynamicist.

In order to accurately predict such a flow numerically, the instability mechanisms present in the incompressible flow, and any additional mechanisms present in the compressible flow, must be accurately resolved both spatially and temporally. Compressible phenomena, such as shocklets, must be accurately predicted in order to understand their interaction with vortices and their effect on instability mechanisms. The effects of assumptions made in an analysis on the accuracy of resolving the instability mechanisms must be addressed. Because of the sensitive nature of instabilities to disturbances, errors made in the analysis may be amplified to a point where they affect the resulting flow physics. The standard zero normal pressure gradient boundary condition used on solid surfaces in compressible viscous flow calculations such as those performed by Visbal (1990) and Choudhuri and Knight (1995) may have a significant effect on the resulting flow evolution, because the eruption event described above is characterized by a region of high normal pressure gradient relative to the wall. Since the unsteadiness is of prime interest in dynamic stall flows, time-accuracy of the analysis is of fundamental importance.

The objective of this study is to: i) develop an analysis that is applicable to dynamic motion CFD problems, ii) to use object-oriented programming techniques in the development of the analysis to facilitate the use of complex CFD techniques (such as flow-adaptive grids, Chimera grids, etc.) on ever changing high-performance massively parallel computer platforms, and iii) to develop simulation results for dynamic-stall phenomena under a variety of flow conditions to help in the understanding of the role compressibility plays in the phenomenon of dynamic stall.

Underlying the entire analysis are the physical issues of how to accurately capture instabilities and avoid error growth, as well as the computational issues of optimization and

efficiency. The computational issues should not be allowed to affect the analysis in such a way as to lower confidence in the resulting solutions through the addition of errors.

Analysis and Discussion

The analysis begins with the first principles of classical field theory applied to a continuum fluid. The integral equations are transformed into a set of non-linear partial differential equations written in terms of an arbitrarily moving coordinate system. Unsteadiness of the coordinate system as well as that of the bodies in the flow are accounted for in this manner. Cross derivatives are retained in the resulting differential equations to ensure accuracy of the solutions of these equations. The equations are neither modified by explicitly adding damping terms nor by using a turbulence model. Both of these are inconsistent with the very equations we are trying to solve. A consistent set of boundary conditions have been developed which removes the requirement for a pressure boundary condition on solid surfaces. Consistent far-field conditions are used, unlike the zeroth-order extrapolation or characteristics-based boundary conditions commonly used.

The differential equations are discretized, resulting in a non-linear algebraic equation set whose solution approaches the continuum solution at a known rate. Based on the results of a model problem, it can be said that, as the grid is refined, the physics of the discrete equation set does approach that of the continuum equations. *The discrete equations are solved at all points within the flow*, including pseudo-boundaries introduced by computational topologies such as branch cuts and multi-block/Chimera-grid boundaries. Time accuracy is assured by complete convergence of the residual of the non-linear equation set, guaranteeing its solution, and through the use of a fully implicit time discretization. Errors introduced through incomplete convergence of the equations may adversely affect the flow physics of the discrete equation set and, thus, the instability events that are being captured.

Numerical Jacobians are used to efficiently generate the resulting linearized coefficient matrix. A fully direct method is not feasible for solving the resulting very large system of equations, even when the sparse nature of the system is considered. Thus, an iterative method is used to drive the residual of the equation set to zero at each time step.

Computational optimization pervades every aspect of a simulation of a challenging flow problem. To lower memory requirements, the number of operations and numbers of iterations performed, flow-adaptive grid techniques are used to achieve near-optimal grid-point placement and, more importantly, to accurately capture unsteady flow phenomena. Multi-grid techniques have been programmed and will be used, if necessary, to accelerate the convergence of the unsteady equations at each time step. If the problem requires the use of a direct method for convergence-related issues, then a multi-block 'divide and conquer' approach is planned to break the problem into smaller pieces. In addition, underlying all numerical computations is the issue of matching algorithms to the machine architecture in use. The use of massively parallel machines is a necessity for computing dynamic motion CFD problems. Because of the need to change algorithms when switching architectures, code design becomes important, to reduce the time spent porting between various machines.

Object-oriented programming (OOP) techniques have, therefore, been used to address many of these optimization issues. Code complexity increases dramatically as complex CFD techniques, such as flow-adaptive and multi-block grids, are added to an analysis. Advanced programming techniques are necessary to manage such complexity in an organized manner. The interaction between various codes, such as a grid generator and a flow solver, must be expressed in the program itself, in order to allow effective communication. With the use of OOP languages, portability issues are more directly addressed and more adequately resolved. The use of an object-oriented language actually facilitates efficient programming rather than hindering it, since the way an object is used is decoupled from the

particular way it was implemented on a given machine. Specific optimized algorithms can be used where appropriate, without complicating other parts of a code. A good example of this is linear algebra programming, which forms the basis of all CFD codes. A particular efficient implementation of a matrix operation can be used, where one was not before, without changing the way the CFD program loads up its coefficient matrix. An example of the high performance attainable on a CRAY Y-MP with C++ is given in Fig. 4. Because of the design of the linear algebra objects used by the CFD code, this optimized block matrix routine was used without changing a single line of code in the CFD program.

The use of user-defined types allows the program to be written in terms of objects closer to the problem at hand and allows the compiler to know when, for example, a grid was misused and mark the line with an error flag. In a procedural language, such as FORTRAN, the compiler has no way of knowing if a programmer really meant to pass down to a routine the x-coordinates of a grid first, then the y-coordinates, or vice-versa. All it knows is that it was expecting two arrays, and is ignorant of what they were supposed to be representing.

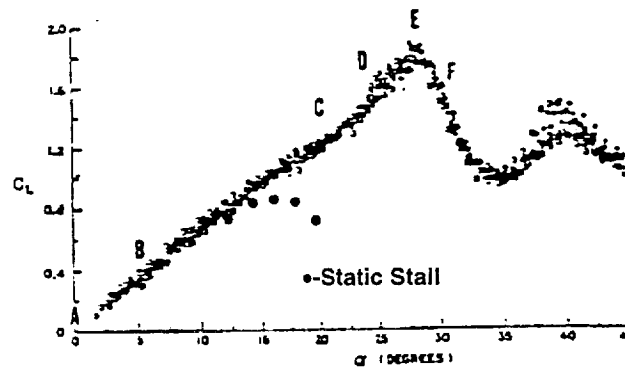
Summary

The compressible flow past a maneuvering airfoil is being studied. Currently, issues related to boundary conditions, including proper treatment of branch cuts and multi-block/Chimera boundaries are being addressed. Numerical analysis issues for solving large-scale iterative problems are being examined. Finally, object-oriented programming

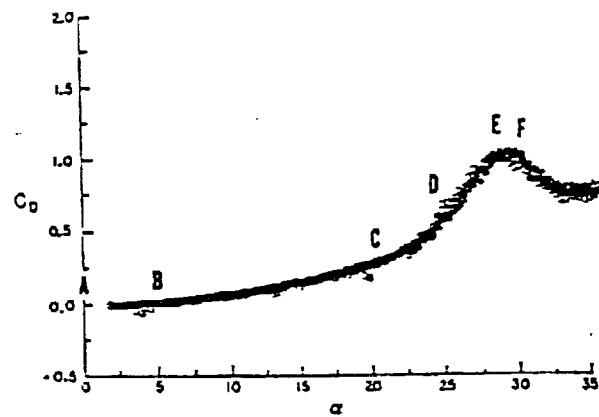
is being used to facilitate the incorporation of complicated CFD techniques into flow codes as well as to address issues of software portability on high-performance supercomputers. Results of this effort, which were not ready at the time of the Conference, will be disseminated shortly in the form of a report.

References

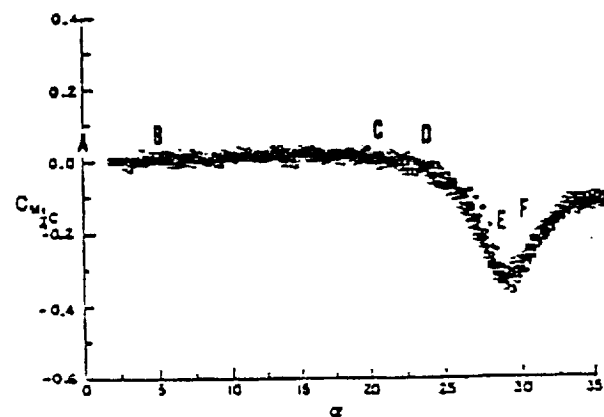
- Chandresekhar, M. S., Carr, L. W., and Wilder, M. C., "Interferometric Investigations of Compressible Dynamic Stall Over a Transiently Pitching Airfoil," *AIAA Paper* 93-0211, January, 1993.
- Choudhuri, P. G. and Knight, D. D., "Effects of Compressibility, Pitch Rate and Reynolds Number on Unsteady Incipient Boundary Layer Separation Over a Pitching Airfoil," *AIAA Paper* 95-0782.
- Ghia, K. N., Yang, J., Osswald, G. A., and Ghia, U., "Study of the Role of Unsteady Separation in the Formation of Dynamic Stall Vortex," *AIAA Paper* 92-0196.
- Visbal, M. R., "Dynamic Stall of a Constant-Rate Pitching Airfoil," *Journal of Aircraft*, Vol. 27, p. 400, May, 1990.
- Yang J., Ghia, K. N., Ghia, U. and Osswald, G. A., "Management of Dynamic Stall Phenomenon Through Active Control of Unsteady Separation," *AIAA Paper* 93-3284.



a) Lift History

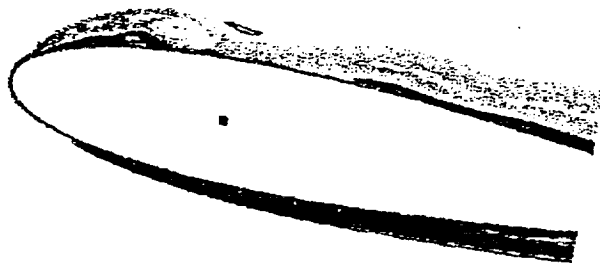


b) Drag History

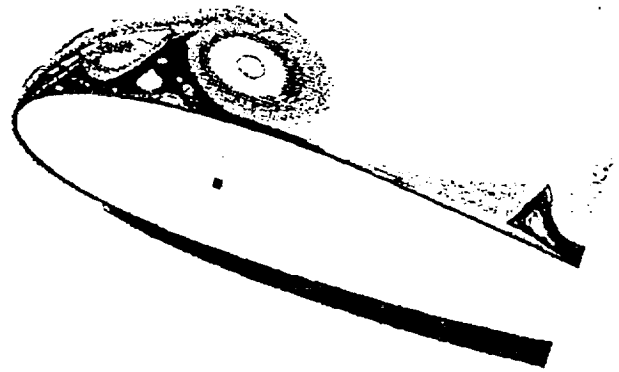


c) Moment History

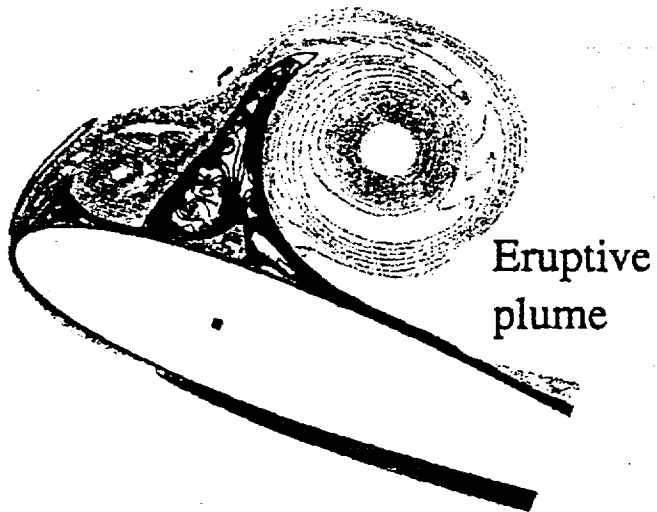
Figure 1.-Dynamic Stall: Lift, Drag and Moment Curves of Jumper, et al., (1988)



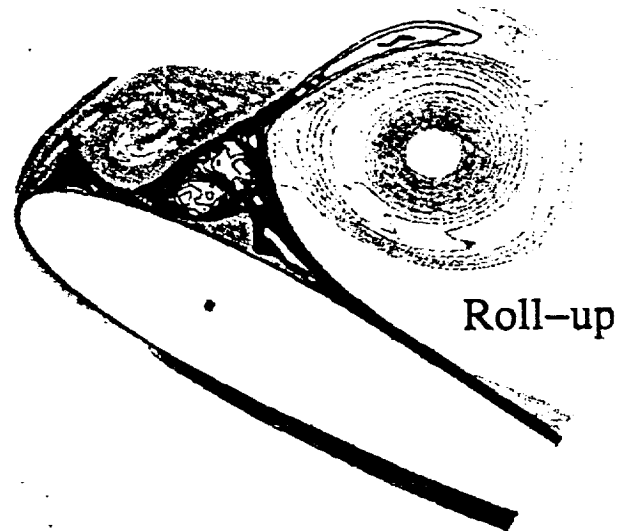
a) Initial leading edge separation



b) Onset of interaction, sharp local pressure changes

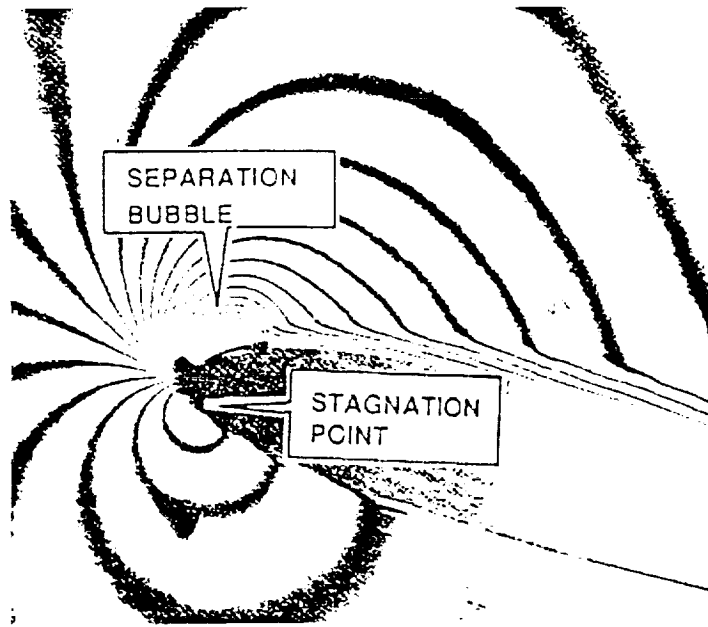


c) Strong interaction

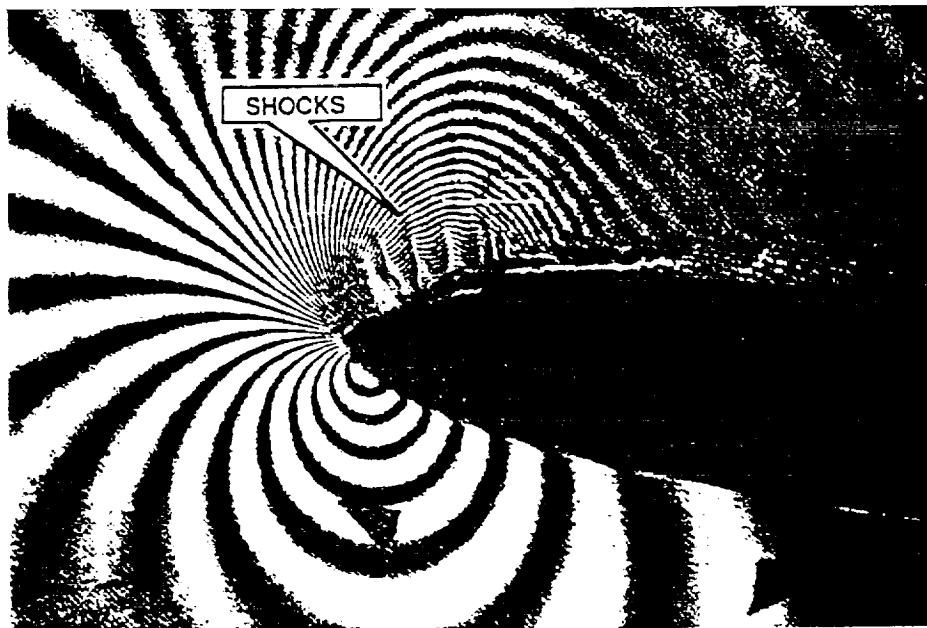


d) Inviscid Interaction

Figure 2.-Vortex Structure in Sequence of Events for Dynamic Stall (K. Ghia, et al., 1992)



a) $M=0.3$, $\alpha=12^\circ$, $\alpha^+=0.03$



b) $M=0.45$, $\alpha=12.6^\circ$, $\alpha^+=0.0313$

Figure 3.-Compressibility Effects on Dynamic Stall (Chandrasekhara, et al., 1993)

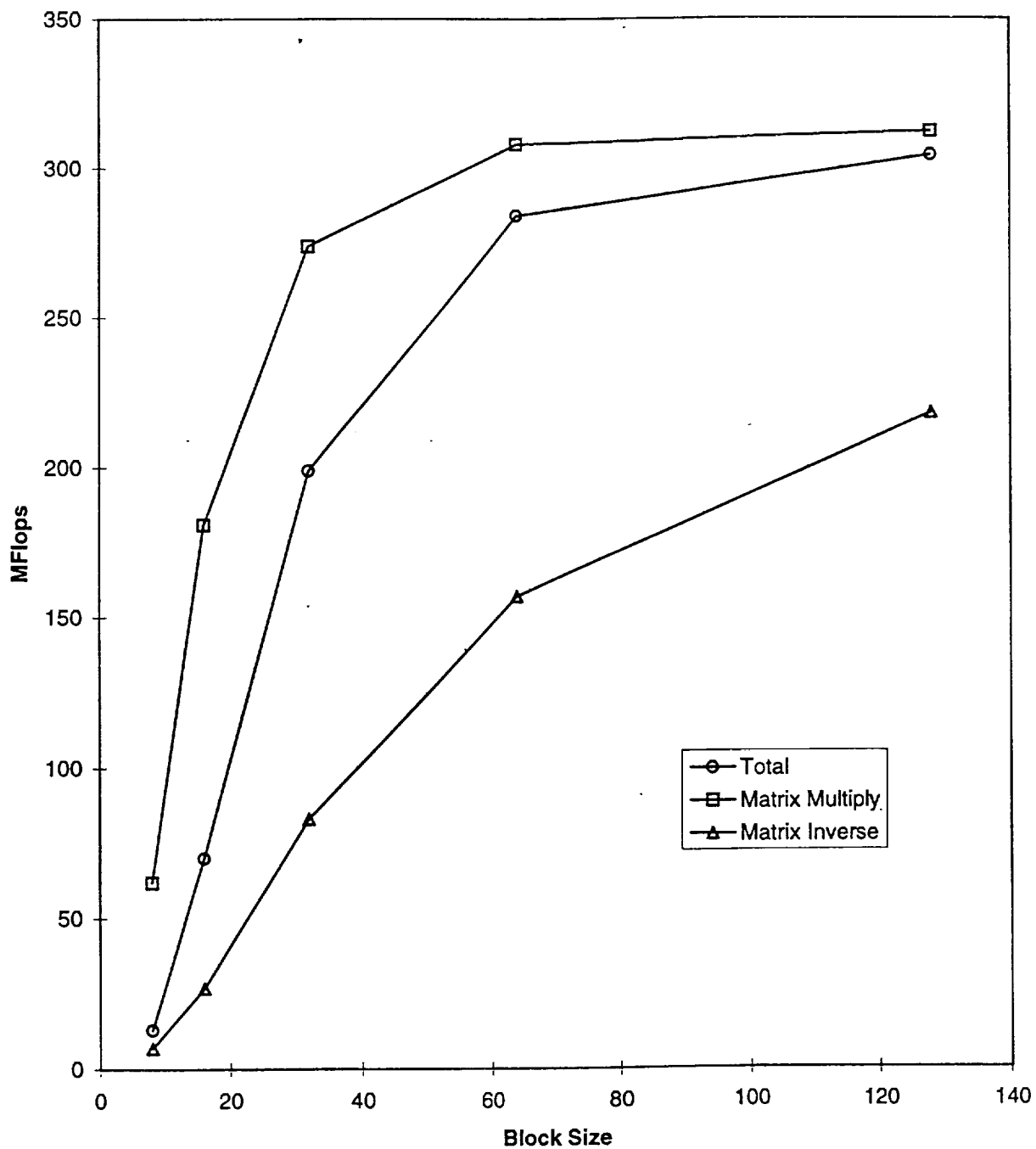


Figure 4.-Cray Y-MP Performance for C++ Block Matrix Inverse